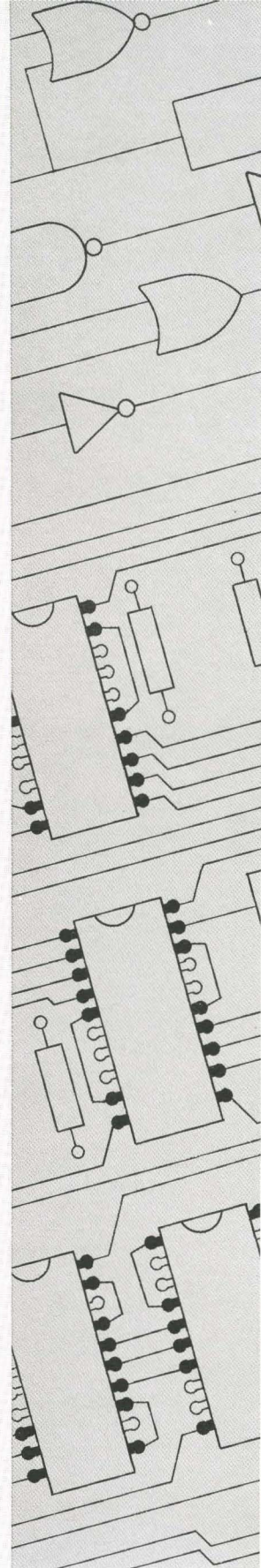


PCB SYSTEM OVERVIEW

PRINTED CIRCUIT BOARD
DESIGN SYSTEM

000-0097-02

p-cad[®]
PERSONAL CAD SYSTEMS INC.



PCB SYSTEM OVERVIEW

August 1986

000-0097-02

Personal CAD Systems, Inc.
1290 Parkmoor Avenue
San Jose, CA 95126
(408) 971-1300

COPYRIGHT

Copyright (c) 1986 by Personal CAD Systems, Inc. (P-CAD).

All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise, without the prior written permission of Personal CAD Systems, Inc.

Personal CAD Systems, Inc. provides this manual "as is" without warranty of any kind, either expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. P-CAD may make improvements and/or changes in the product(s) and/or the program(s) described in this manual at any time and without notice.

Although P-CAD has gone to great effort to verify the integrity of the information herein, this publication could contain technical inaccuracies or typographical errors. Changes are periodically made to the information herein. These changes will be incorporated in new editions of this publication.

TRADEMARKS

P-CAD, PC-CAPS, PC-CARDS, PC-LOGS, PC-BACK, PC-DRC/NLC, PC-DRILL, PC-FORM, PC-PLACE, PC-LINK, PC-MODEL, PC-NODES, PC-PACK, PC-PHOTO, PC-PLOTS, PC-PRINT, PC-ROUTE, POSTSIM, PREPACK, and PRESIM are trademarks of Personal CAD Systems, Inc. (P-CAD).

ABOUT THIS MANUAL

This manual is designed to serve as an overview of P-CAD's PCB-1, PCB-2, and PCB-3 systems and as a guide to using all the programs in your system together. The PCB systems enable you to produce schematics (PCB-2 and PCB-3 only), printed circuit board designs, plots, and listings of components and nets. To learn about using your PCB system, read this manual before reading any of the individual program manuals.

Chapter 1, INTRODUCTION, gives instructions for using the PCB systems' documentation to learn to operate your system and lists system configuration requirements.

Chapter 2, SYSTEM DESCRIPTION, gives an overview of the functions of the PCB systems and the programs that perform them.

Chapter 3, USING YOUR PCB SYSTEM, provides information on how to work with the systems, including file handling and system concepts.

The appendixes provide further information about the PCB system for your reference.

NOTATION

This manual gives step-by-step procedures and examples. To make it easy for you to follow these procedures, we use the following notation.

<xxxx> Angle brackets around lowercase letters indicate a variable name that may be entered by the system or by you. For example:

`<filename>.SCH.`

[] Square brackets indicate the name of a key. For example:

`[Return].`

[Return] `[Return]` indicates the key that is used to execute a command or accept an option. This key may be labeled differently, depending on your system. For example: `[RETURN]`, `[↵]`, `[Enter]`, `[Enter ↵]`, `[ENTER]`.

[-|] Square brackets connected with a hyphen indicate keys that must be pressed simultaneously. For example:

Press `[Ctrl]-[Alt]-[Del]`.

UPPER Uppercase letters indicate a command or an element that must be typed as shown. For example:

Type `PCPLOTS` and press `[Return]`.

/ A forward slash begins a keyboard command. For example:

/EXE.

* An asterisk in a filename or in a filename extension indicates that any character(s) can occupy that position and all the remaining positions in the filename or extension. For example, the DOS command

DIR *.SYM

displays a list of all the filenames with the extension .SYM in the current directory.

TESTFILE TESTFILE is a sample filename, which you must replace with the filename you intend to use. For example:

Database Filename :TESTFILE.SCH
Netlist Filename :TESTFILE.NLT

CONTENTS

CHAPTER 1. INTRODUCTION.....	1-1
Learning Your PCB System.....	1-1
System Configuration.....	1-1
CHAPTER 2. SYSTEM DESCRIPTION.....	2-1
Schematic Capture.....	2-1
Creating Symbols.....	2-2
Creating Schematics.....	2-3
Hierarchical Structure.....	2-4
Multisheet Schematics.....	2-5
Producing Netlists.....	2-7
Printing and Plotting the Schematic.....	2-8
Packaging Components.....	2-8
Physical PCB Design.....	2-10
Creating Parts.....	2-11
Designing the PCB.....	2-12
Creating a PCB from Parts.....	2-12
Packaging a Schematic into a PCB.....	2-12
Checking the PCB Design.....	2-13
Back-Annotating the Schematic.....	2-14
Producing Netlists.....	2-14
Printing and Plotting the PCB Design.....	2-15
PCB Systems Data Flow.....	2-15
Input/Output File Summary.....	2-18
CHAPTER 3. USING YOUR PCB SYSTEM.....	3-1
Using the Programs.....	3-1
P-CAD Directory Structure.....	3-1
Changing Directories.....	3-2
Indicating Paths.....	3-3
Viewing and Printing ASCII Files.....	3-4

CONTENTS (Continued)

System Concepts	3-6
Grid Structure	3-6
Layer Structure	3-10
Backing Up Your Data Files	3-11
Procedures for End-to-End Schematic to PCB Design	3-12
APPENDIX A. PCB SYSTEM MATERIALS	A-1
APPENDIX B. P-CAD FILENAME EXTENSION CONVENTIONS	B-1
APPENDIX C. LOADABLE DEVICE DRIVER FILES	C-1
The Loadable Driver Reference File	C-1
System Driver Files	C-3
Display Device Driver Files	C-4
Input Device Driver Files	C-9
INDEX	I-1

FIGURES

2-1.	PC-CAPS Symbol: NAND Schematic Component	2-2
2-2.	PC-CAPS Schematic.	2-3
2-3.	Hierarchical Schematic Structure.	2-5
2-4.	Multisheet Schematic Structure	2-6
2-5.	Packaged PCB Database	2-10
2-6.	PC-CARDS Part: 74368A Dip Component.	2-11
2-7.	Routed PCB Database	2-13
2-8.	PCB Systems Flowchart.	2-16
3-1.	P-CAD Recommended Directory Structure.	3-2
3-2.	Grid Scale of P-CAD Library Schematic Symbols	3-8
3-3.	Grid Scale of P-CAD Library Parts.	3-9

TABLES

2-1.	Input/Output File Summary.	2-18
3-1.	ASCII Output Files	3-5
B-1.	PCB Filename Extension Conventions.	B-1
B-2.	PC-PRINT, PC-PLOTS, and PC-PHOTO Filename Extension Conventions	B-3

CHAPTER 1. INTRODUCTION

This manual introduces you to Personal CAD (P-CAD) System's PCB-1, PCB-2, and PCB-3 systems for computer-aided printed circuit board design. The PCB systems allow you to design printed circuit boards, which you can then edit, produce listings from, plot, and photoplot. The PCB-2 and PCB-3 systems allow you to build schematics, which you can then package into printed circuit boards. PCB-3 provides additional tools to help with placement and routing. P-CAD also provides interfaces to a number of other systems for logic simulation and printed circuit board layout.

LEARNING YOUR PCB SYSTEM

Each PCB system consists of several programs that function together to provide a flexible and powerful system. This manual provides an overview of the PCB systems applications and explains how to use the programs together to perform these functions.

The *P-CAD Installation Guide* contains the procedures for installing the system. The operation of each program is described in the individual program manuals.

All the examples and instructions in this manual and in all the program manuals assume that you have installed the system according to the procedures in the *P-CAD Installation Guide*.

SYSTEM CONFIGURATION

P-CAD's PCB products run under PC-DOS or MS-DOS version 2.0, or greater, on the computer systems and hardware listed below.

Computer Systems:

Your PCB system works on one of the following computer systems.

- IBM PC, PC/XT, PC/AT, or equivalent, with one of the following graphics configurations:

- IBM Color Graphics Adapter and Monitor
- Conographics Color Graphics Adapter and Monitor
- Hercules Monochrome Graphics Adapter and Monitor
- IBM Professional Graphics Controller with Professional Graphics Display
- IBM Enhanced Graphics Adapter with Graphics Memory Expansion (64K RAM) and Enhanced Color Display

- Texas Instruments Professional or Portable with TI color graphics adapter and monitor
- Tandy TRS-80 Model 2000 with Tandy color support kit and CM1 color monitor

Minimum Hardware:

- 640K RAM
- One 360K floppy disk drive
- 10 Mbyte hard disk drive or shared storage on network
- RS-232 serial port
- P-CAD security device

NOTE: A mouse (Mouse Systems, Logitech, or equivalent) is highly recommended.

These requirements are for the whole system. You can, if you want, use a separate, smaller computer system for printing or plotting. PC-PLOTS, PC-PRINT, and PC-PHOTO do not require a hard disk for program operation. PC-PLOTS and PC-PHOTO require only 256K of memory, and PC-PRINT requires 512K.

P-CAD's PCB products use driver files to configure the P-CAD software to your computer hardware. When you run the INSTALL program as described in the *P-CAD Installation Guide*, you specify your computer hardware, mouse, and graphics display controller.

The P-CAD INSTALL program creates a loadable driver reference file (PCADDRV.SYS) containing three lines that specify the name of the driver files (<filename>.DRV) for your CPU, graphics monitor, and input device type. Filenames with a .DRV extension are device driver files, which the AUTOLOADER program copies into the \PCAD\DRV directory.

If your system does not already have a loadable driver reference file containing the appropriate driver files for your system, you must create one. Refer to Appendix C, "Loadable Device Driver Files," for more information.

PCB SYSTEM OVERVIEW 1-4

CHAPTER 2. SYSTEM DESCRIPTION

Each PCB system consists of a comprehensive set of software programs for electronic design applications. All three systems allow you to design printed circuit boards using the PC-CARDS program and to perform a wide variety of related functions using several utility programs. The PCB-2 and PCB-3 systems also provide the capability for schematic capture, using the PC-CAPS program, and allow you to package the schematic components and connectivity into a PCB design that you can then complete using PC-CARDS and the related utilities. The PCB-3 system provides, in addition, automatic placement and automatic routing, using the PC-PLACE and PC-ROUTE programs.

The PCB system stores all information in files. According to DOS conventions, each file has a name of up to eight characters, followed by a period and a filename extension of up to three characters. When you start a design or enter other information, you assign it a filename. P-CAD uses certain conventions (described in Appendix B, "P-CAD Filename Extension Conventions") for filename extensions. As you work with a design, you produce new files that generally have the same filename with different extensions.

This chapter gives an overview of the PCB systems. It explains the applications of the systems, schematic capture and physical PCB design, and how the programs work together to perform these applications. Information on the operation of each program is in the corresponding user's manual.

SCHEMATIC CAPTURE

To design a schematic, you use the PC-CAPS program. PC-CAPS is one of the two central programs of the PCB-2 and PCB-3 systems. It enables you to build schematics using predefined symbols and to create symbols. It allows you to design hierarchical and multisheet schematics. PC-CAPS features color graphics, a menu of commands for easy entry of graphics and component data, and a flexible grid structure.

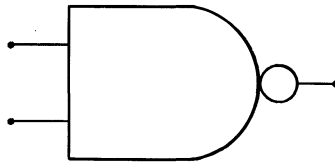
To create a schematic in PC-CAPS, you place symbols representing schematic components on the design and connect them together to form the desired circuit. To help you use this technique for designing schematics, PC-CAPS has two operating modes, symbol mode for creating symbols and detail mode for creating schematics.

The following sections explain how to create symbols and schematics, produce netlists, printouts, and plots, package schematic components into physical parts, and back-annotate the schematic.

Creating Symbols

Symbols are created in PC-CAPS symbol mode. Each symbol consists of a graphic drawing, information about component pins, and other relevant information. Each component is stored in a file with a .SYM filename extension.

You can order one or more libraries of standard component symbols with your PCB-1, PCB-2, or PCB-3 system. You can use these P-CAD library symbols or you can edit them for your purposes. You can also create your own symbols as needed. Figure 2-1 shows a sample symbol.

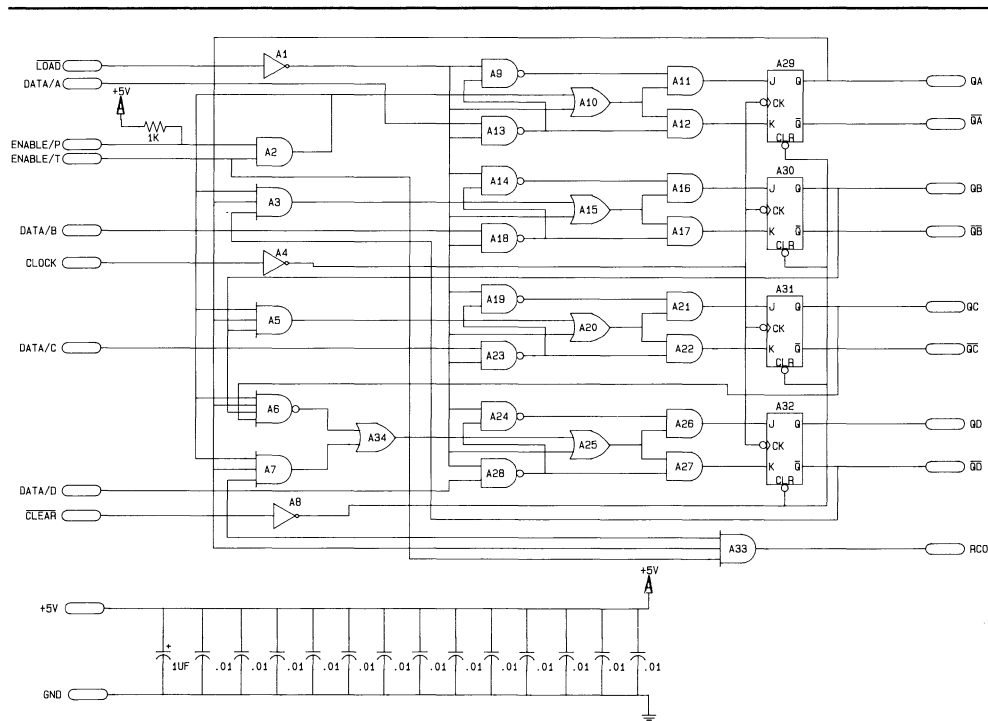


85024

Figure 2-1. PC-CAPS Symbol: NAND Schematic Component

Creating Schematics

You create schematics in PC-CAPS detail mode by accessing the files for the required symbols, placing the symbols in the desired positions, and adding wires to connect the components. You can edit the schematic as needed. You store the schematic database in a file with a .SCH filename extension. Figure 2-2 shows a sample schematic database.



85025

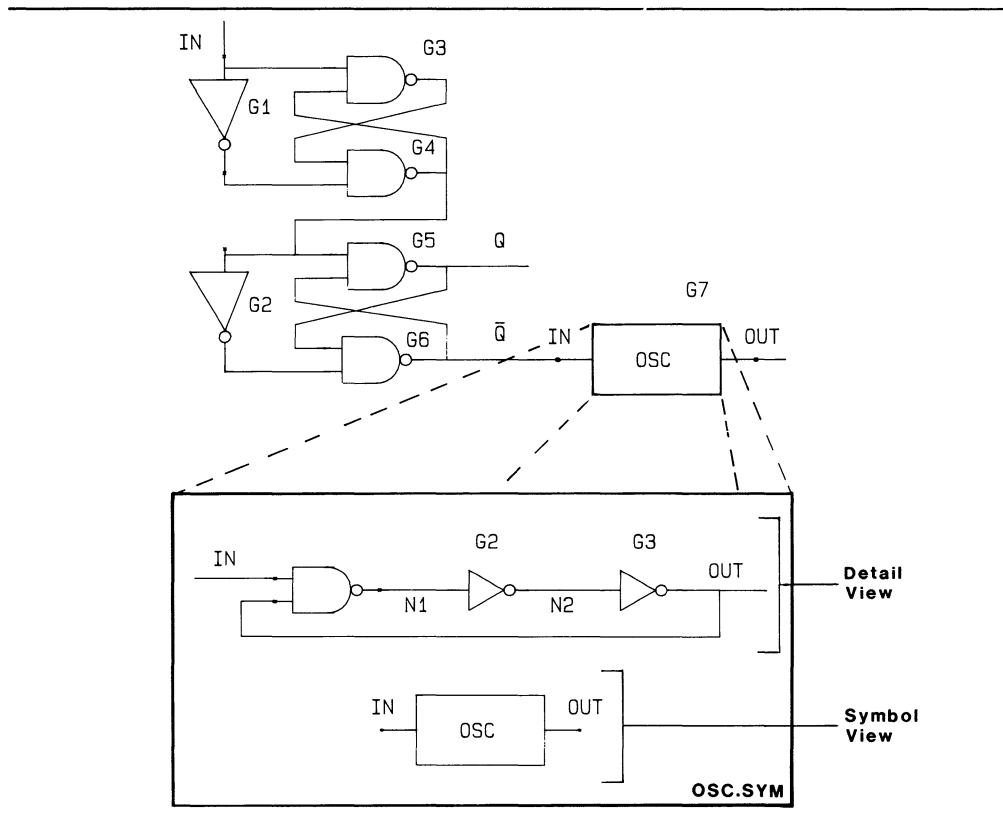
Figure 2-2. PC-CAPS Schematic

Many schematics require more detailed information than is convenient to store in one file. For such schematics, PC-CAPS allows you to use hierarchical and multisheet structures, which are described below.

Hierarchical Structure

Hierarchical structure is the use of symbols representing subcircuits. After you create a schematic, you can use PC-CAPS symbol mode to create a symbol that represents the schematic. Both the detail and symbol views of the schematic are present in the same schematic database file. You can then use the symbol as a component in another schematic.

The PCB system supports up to 15 levels of hierarchy. Figure 2-3 illustrates hierarchical structure.

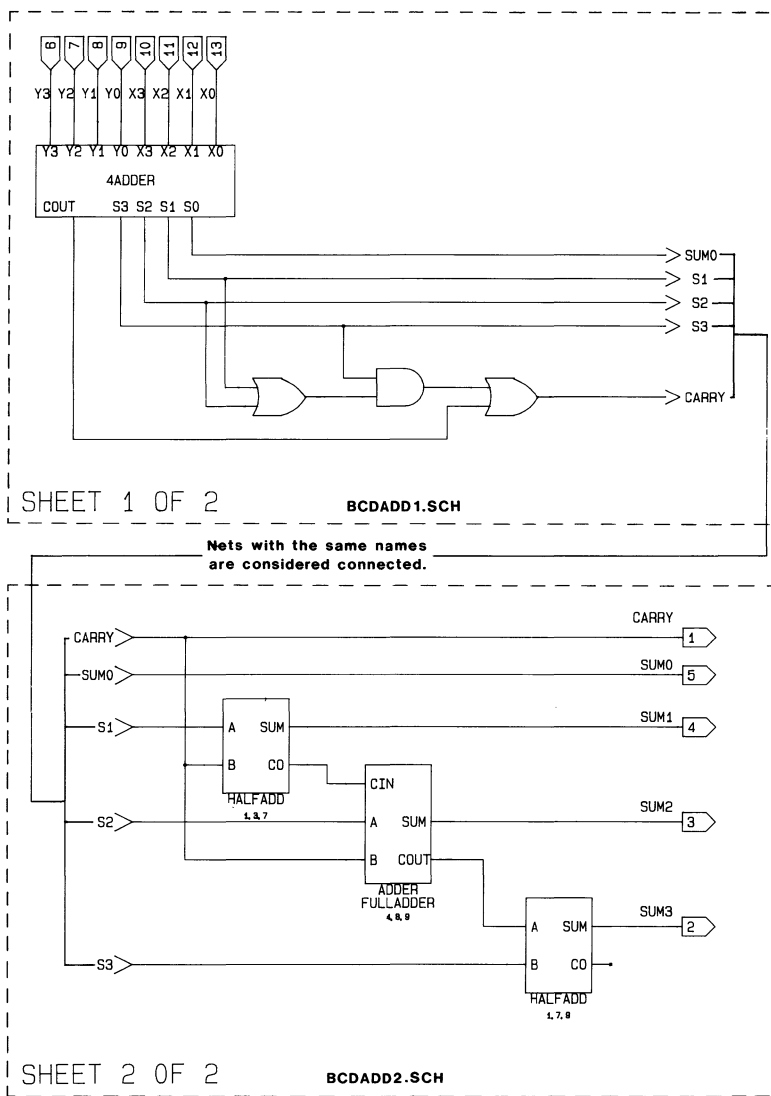


85026

Figure 2-3. Hierarchical Schematic Structure

Multisheet Schematics

PC-CAPS allows you to create multisheet designs, which consist of several schematic drawings, stored as separate files, that are interconnected. You specify the interconnections by giving connected wires the same name on each sheet. Figure 2-4 illustrates multisheet structure.



85027

Figure 2-4. Multisheet Schematic Structure

Producing Netlists

You can use the PCB system to produce netlists that contain component and circuit connectivity information for a schematic. The PCB system produces netlists in two formats:

- Binary format is readable by the system and is used to transfer data about the schematic to other P-CAD programs, including P-CAD interfaces to other systems.
- ASCII, or "human readable" format, is used for printouts of netlist information.

You must extract a binary netlist before you can produce ASCII lists.

To produce a binary netlist of your schematic database, you input the schematic file to the PC-NODES utility program. PC-NODES extracts the netlist, which lists the schematic components and their interconnections, and outputs it in a file with a .NLT filename extension.

Binary netlists extracted from hierarchical or multisheet schematic databases must also be flattened and/or connected into an expanded netlist. To link the netlists, you input them into PC-LINK, which combines them into an expanded netlist file with a .XNL filename extension.

To convert a binary netlist into ASCII format, you use the PC-FORM utility program. PC-FORM reads binary netlists produced by PC-NODES, PC-LINK or PC-PACK and outputs ASCII component, node, wire, packaging, and/or material lists for the schematic or packaged database.

Printing and Plotting the Schematic

To produce printouts and plots of a schematic, the PCB systems provide three utilities that read schematic plot files (<filename>.PLT) produced by PC-CAPS and output plotting instructions to printers and plotters. PC-PRINT allows you to produce dot-matrix printouts. PC-PLOTS allows you to produce pen plots, and PC-PHOTO provides photoplotting capability.

Packaging Components

With the PCB-2 and PCB-3 systems, you can use the information from a PC-CAPS generated schematic database to produce a packaged and electronically connected PCB database which you can edit with the PC-CARDS program. You can then feed back packaging information and pin swaps, component swaps, or name changes made in this PCB database to PC-CAPS to update the original schematic database.

To prepare the schematic for packaging, you must extract a binary netlist from the schematic using the PC-NODES and, if necessary, the PC-LINK program.

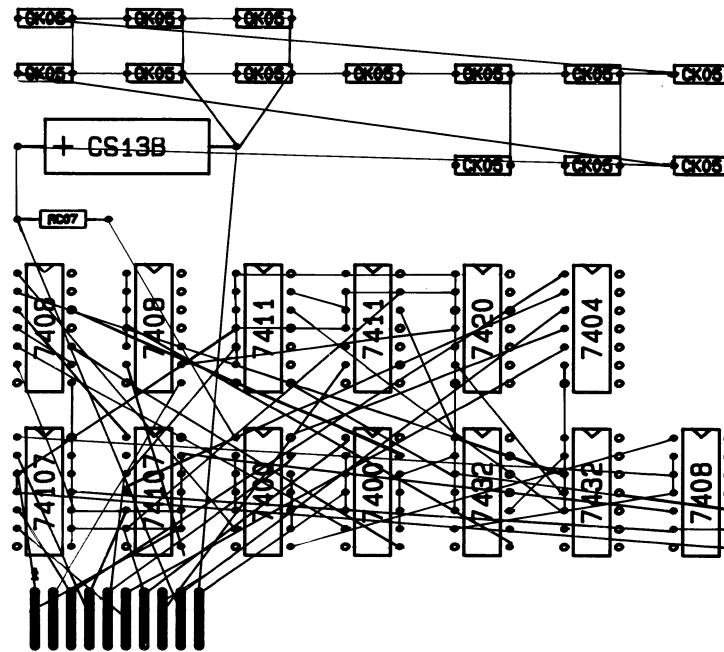
You also need an ASCII component library cross-reference list that maps all the PC-CAPS symbols used on the schematic onto functionally equivalent PCB parts that have been created using the PC-CARDS program. This list is on a file with a .FIL extension. You can create this cross-reference list using a text editor or use the list supplied with your P-CAD libraries. You input this list into the PREPACK program, which translates it into binary format and gives it a .LIB extension.

To package the components in a schematic into physical parts on a printed circuit board, you use the PC-PACK program. You input the netlist file (<filename>.NLT or <filename>.XNL), the binary component library cross-reference file (<filename>.LIB), and a PC-CARDS generated PCB layer database, which contains starting settings for layers, the grid, and line width.

PC-PACK reads this input and maps the logical components from the schematic database onto functionally equivalent packaged components in the PCB database.

PC-PACK outputs the following:

- A new PCB database (<filename>.PCB) with schematic components packaged into physical parts and connected (via "ratsnest lines") in a preliminary arrangement (shown in Figure 2-5).
- A binary-format packaged netlist (<filename>.PNL). You can input this file into PC-FORM if you want to print netlists for the newly-packaged PCB database.
- A packaging command file (<filename>.CMD) that contains symbol-to-part mapping for back annotation to the original schematic database. You can input this file directly into PC-CAPS if you want to produce schematic netlists or plots that include the packaging information. Alternately, you can input this file into PC-BACK together with changes you make while editing the PCB database, as explained later in this chapter in the section "Back-Annotating the Schematic."



85032

Figure 2-5. Packaged PCB Database

PHYSICAL PCB DESIGN

To design a printed circuit board, you use the PC-CARDS program. PC-CARDS is the central program of the PCB-1 system and, with PC-CAPS, one of the two central programs of the PCB-2 and PCB-3 systems. It enables you to lay out a PCB database using predefined graphic "parts" representing physical components, and it checks connectivity to help eliminate potential design errors. PC-CARDS features color graphics, a menu of commands for easy entry of graphics and component data, and a flexible grid structure.

To create a PCB database in PC-CARDS, you place graphic parts on the design and connect them together to form the desired circuit. If you have the PCB-2 or PCB-3 system, you can automatically package your schematic design into logically equivalent parts with full circuit connectivity.

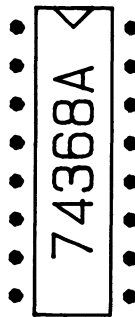
PC-CARDS has two operating modes, one for creating parts (symbol mode) and one for PCB design layout (detail mode).

The following sections explain how to create parts and PCB databases, check your design, back-annotate your schematic, and produce netlists, printouts and plots.

Creating Parts

Parts are created in PC-CARDS symbol mode. Each part consists of a graphic drawing, information about component pins, and other relevant information. Each part is stored in a file with a .PRT filename extension.

Your PCB system includes one or more libraries of standard parts. You can use these parts as they are or you can edit them for your purposes, and you can create additional parts as needed. Figure 2-6 shows a sample part.



85033

Figure 2-6. PC-CARDS Part: 74368A Dip Component

Designing the PCB

You create and edit PCB databases in PC-CARDS detail mode. You can create a PCB in either of the two ways described below.

Creating a PCB from Parts

You can access part files in PC-CARDS detail mode and place the parts on a PCB, add wires to connect the parts, and edit the PCB as needed.

Packaging a Schematic into a PCB

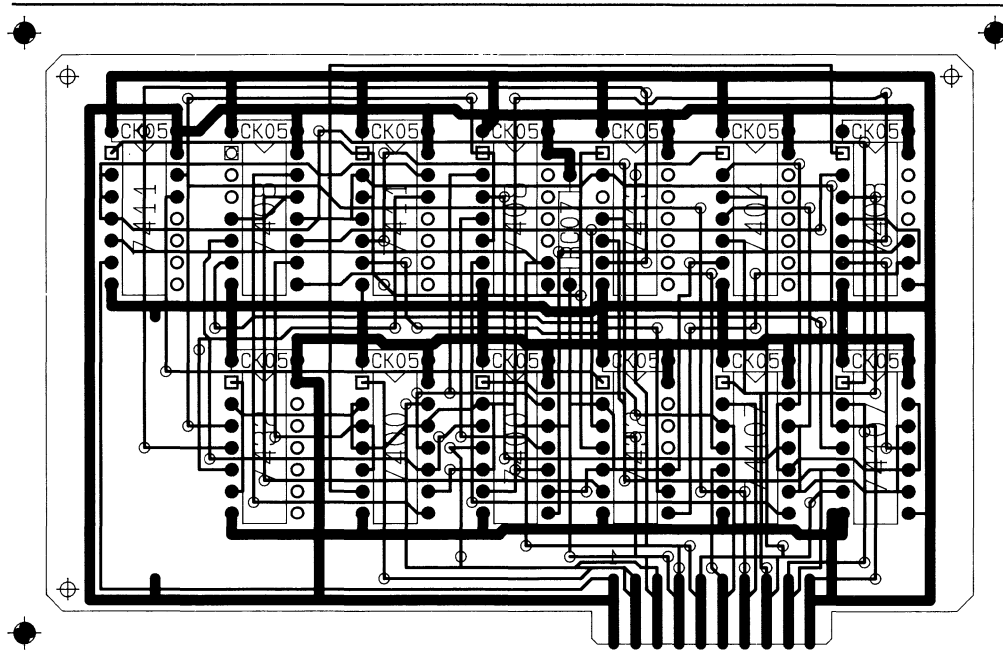
With the PCB-2 or PCB-3 system, you can construct a schematic using PC-CAPS, then use the utilities described in the previous section to package it into a PCB database. You can edit and route the PCB database as needed using PC-CARDS.

With PCB-3, before using PC-CARDS, you can use the automatic placement program to automatically place components and the automatic routing feature to enter traces.

The PC-PLACE automatic placement program includes histogram and force vector analysis tools to allow you to interactively improve the component placement. It operates in an environment similar to that of PC-CARDS, in which you can edit component placement and make other changes. You can prepare the PCB database for autorouting using PC-CARDS or PC-PLACE. If you want to pre-wire traces, you must use PC-CARDS.

The PC-ROUTE autorouting program allows you to specify parameters and rules for routing, then automatically routes traces on the PCB. PC-ROUTE may not complete all the routes, so you then use PC-CARDS to edit the database into its final form.

Figure 2-7 shows the routed PCB database.



85034

Figure 2-7. Routed PCB Database

Checking the PCB Design

The PCB systems allow you to check for violations of physical connectivity and of design rules that you can specify, such as the minimum amount of space between traces. The PC-DRC/NLC program allows you to input the PCB database, specify the design rules, and select the layers to be checked. PC-DRC/NLC generates additional layers, which highlight the errors, and an error report file (<filename>.DRC).

Back-Annotating the Schematic

If you create a PCB database by packaging a PC-CAPS schematic, you may want to update the schematic with the information added during packaging and the changes you make to the database using PC-CARDS or PC-PLACE.

The PC-BACK utility program provides back-annotation to the schematic for packaging and PCB editing changes. PC-BACK reads the packaging command file (<filename>.CMD), which is produced by PC-PACK during packaging, and the swap report file (<filename>.RPT), which is produced automatically by PC-CARDS and PC-PLACE when you swap or rename pins and gates.

PC-BACK outputs the back-annotation file (<filename>.BKA), which you can input to PC-CAPS to update the schematic database file with packaging annotation that matches the new PCB database.

Producing Netlists

You can use the PCB system to produce netlists that contain parts and wiring information for a PCB design. The PCB system produces netlists in two formats:

- Binary format is readable by the system and is used to transfer data about the PCB design to other P-CAD programs.
- ASCII, or "human readable" format, is used for printouts of netlist information.

You must extract a binary netlist before you can produce ASCII lists.

To produce a binary netlist of your PCB database, you input the database file to the PC-NODES utility program. PC-NODES extracts the netlist, which lists the PCB parts and their interconnections, and outputs it in a file with a .PNL filename extension. To convert a binary netlist into ASCII format, you use the PC-FORM utility program. PC-FORM reads the binary netlist and outputs ASCII component, node, wire, packaging, and/or material lists for the PCB database.

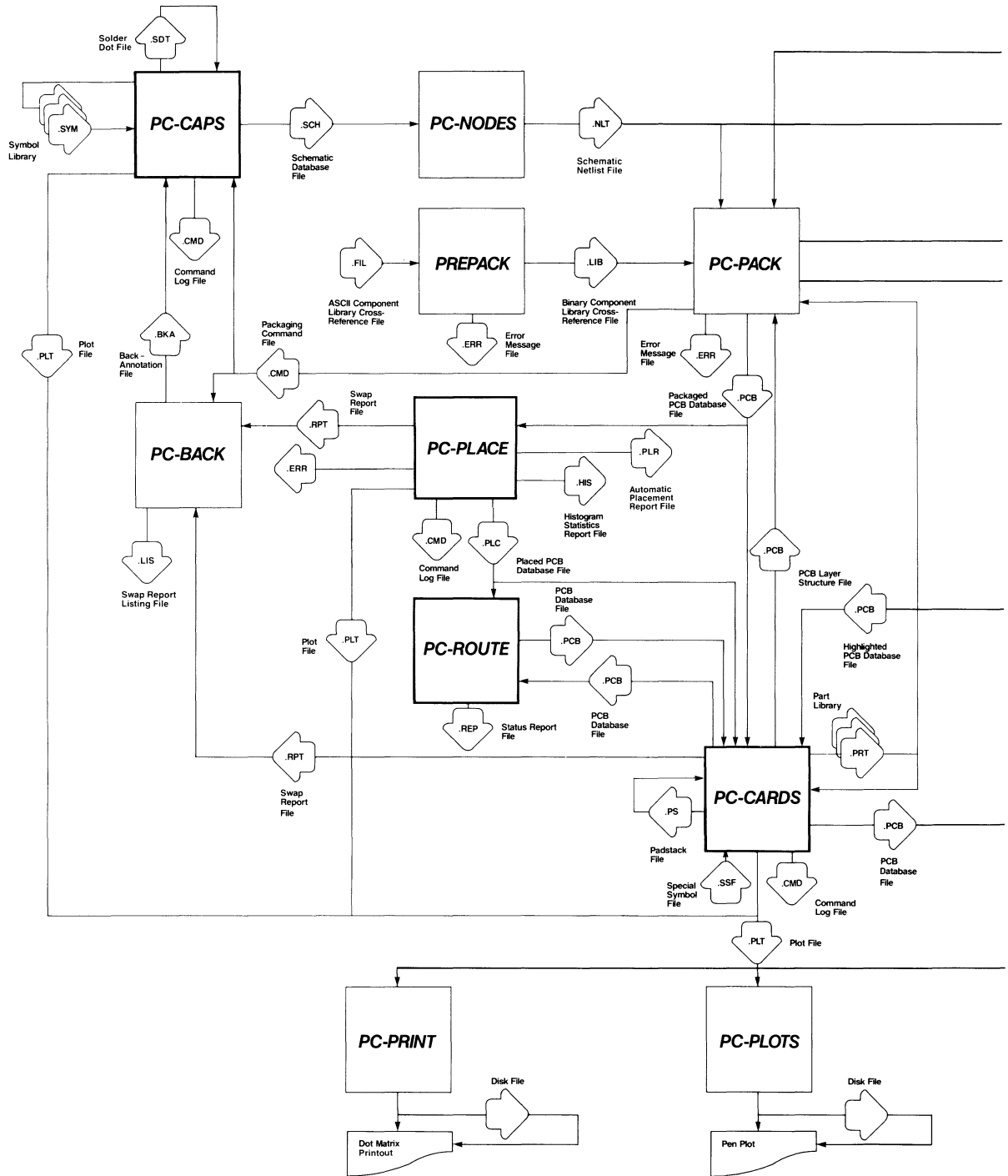
Printing and Plotting the PCB Design

To produce printouts, plots, and photoplots of a PCB design, the PCB systems provide three utilities that read plot files produced by PC-CARDS and output plotting instructions to printers, plotters, and photoplotters. PC-PRINT produces dot-matrix printouts. PC-PLOTS produces pen plots, and PC-PHOTO produces photoplots. To prepare your PCB database for photoplotting, you can add any necessary information, such as padstacks, to the design using PC-CARDS.

PCB SYSTEMS DATA FLOW

Figure 2-8 shows the flow of data among the PCB programs. The poster-size map supplied with the system gives a more detailed view of system operation.

PCB SYSTEM OVERVIEW 2-16



000-0097-02

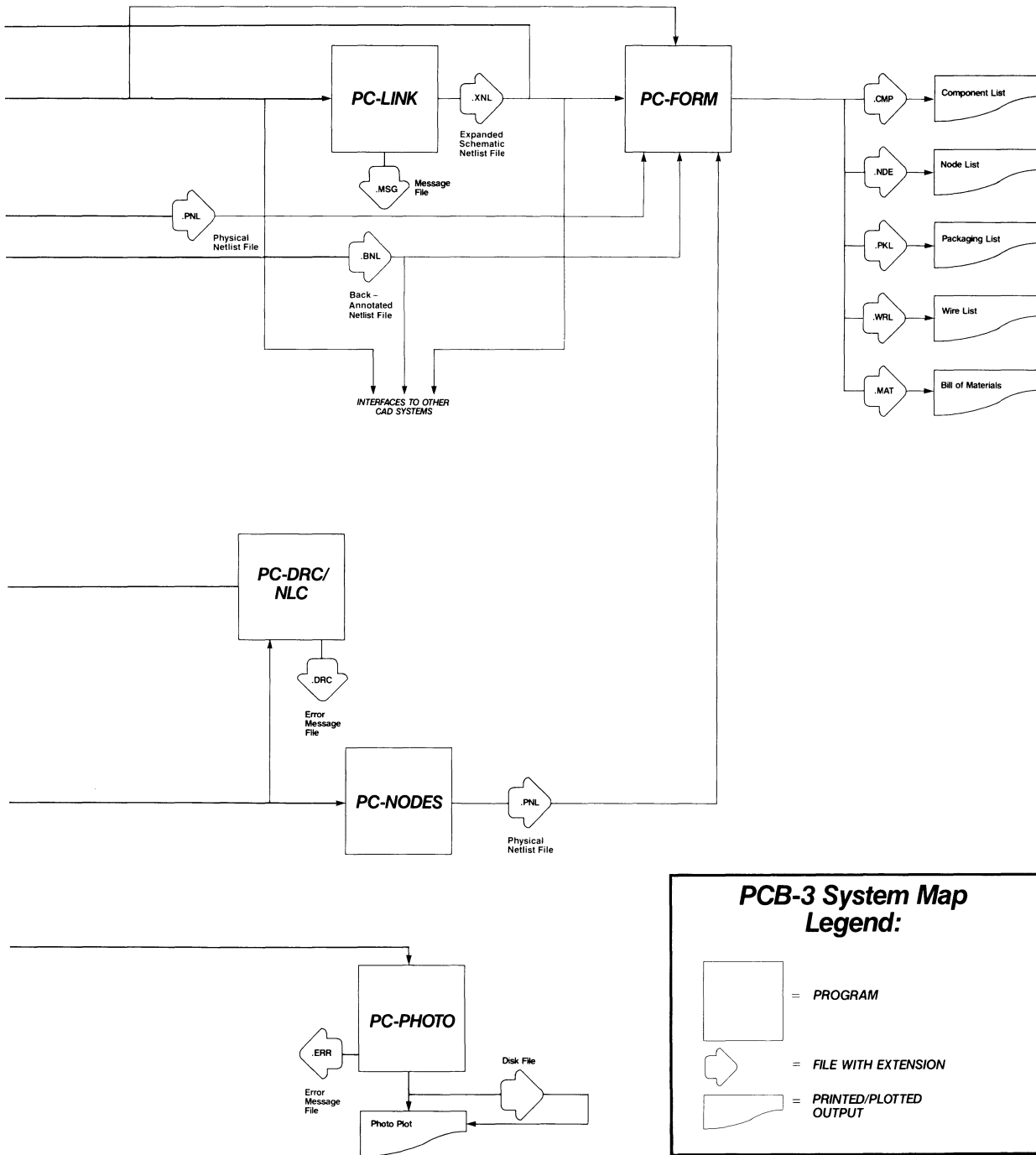


Figure 2-8. PCB Systems Flowchart

INPUT/OUTPUT FILE SUMMARY

Table 2-1 lists the PCB system programs, their input files, and their output files. Preferred or default filenames for each file are in parentheses. Refer to the flowchart in Figure 2-8 and to the system map for more detailed information on the functions of each file.

Table 2-1. Input/Output File Summary

Program	Input	Output
PC-CAPS	Schematic database file (<filename>.SCH)	Plot file (<filename>.PLT)
	Symbol files (<filename>.SYM)	Schematic database file (<filename>.SCH)
	Solder dot file (<filename>.SDT)	Symbol file (<filename>.SYM)
	Back-annotation file (<filename>.BKA)	Solder dot file (<filename>.SDT)
	Packaging command file (<filename>.CMD)	Command log (PCCAPS.CMD)
PC-CARDS	PCB database file (<filename>.PCB)	Plot file (<filename>.PLT)
	Part files (<filename>.PRT)	PCB database file (<filename>.PCB)
	Padstack files (<filename>.PS)	Part file (<filename>.PRT)

Table 2-1 Continued

Program	Input	Output
	Special symbol file (<filename>.SSF)	Padstack file (<filename>.PS) Swap report (<filename>.RPT) Command log (PCCARDS.CMD)
PC-NODES	Schematic database file (<filename>.SCH)	Schematic netlist (<filename>.NLT)
	PCB database file (<filename>.PCB)	Physical netlist (<filename>.PNL)
PC-LINK	Schematic netlists (<filename>.NLT)	Expanded schematic netlist (<filename>.XNL) Message file (PCLINK.MSG)
PREPACK	ASCII component library cross-reference file (<filename>.FIL)	Binary component library cross-reference file (<filename>.LIB) Error message file (PREPACK.ERR)
PC-PACK	Binary component library cross-reference file (<filename>.LIB)	PCB database file (<filename>.PCB)
	Binary netlist (<filename>.NLT) (<filename>.XNL)	Physical netlist (<filename>.PNL)

Table 2-1 Continued

Program	Input	Output
	PCB layer database (<filename>.PCB)	Back-annotated netlist file (<filename>.BNL)
		Packaging command file (<filename>.CMD)
		Error message file (PCPACK.ERR)
PC-PLACE	Packaged PCB database (<filename>.PCB)	Placed PCB database (<filename>.PLC)
		Automatic placement report file (<filename>.PLR)
		Histogram statistics report (<filename>.HIS)
		Swap report (<filename>.RPT)
		Plot file (<filename>.PLT)
		Command log (PCPLACE.CMD)
PC-ROUTE	Placed PCB database (<filename>.PLC)	Routed PCB database (<filename>.PCB)
	PCB database (<filename>.PCB)	Status report file (<filename>.REP)
	Routing strategy file (<filename>.CTL)	

Table 2-1 Continued

Program	Input	Output
PC-FORM	Schematic netlist (<filename>.NLT)	Nodes list (<filename>.NDE)
	Expanded schematic netlist (<filename>.XNL)	Component list (<filename>.CMP)
	Physical netlist (<filename>.PNL)	Packaging list (<filename>.PKL)
	Back-annotated netlist file (<filename>.BNL)	Wire list (<filename>.WRL)
		Bill of materials (<filename>.MAT)
PC-BACK	Packaging command file (<filename>.CMD)	Back-annotation command file (<filename>.BKA)
	Swap report (<filename>.RPT)	Swap report listing file (<filename>.LIS)
PC-PRINT PC-PLOTS	Plot file (<filename>.PLT)	Plotting instructions to printer or plotter
		Plotting instructions disk file
PC-PHOTO	Plot file (<filename>.PLT)	Plotting instructions to photoplotter
		Plotting instructions disk file

Table 2-1 Continued

Program	Input	Output
PC-DRC/NLC	PCB database (<filename>.PCB)	Error message file (PCPHOTO.ERR) Highlighted PCB database (<filename>.PCB) ASCII error report (<filename>.DRC)

CHAPTER 3. USING YOUR PCB SYSTEM

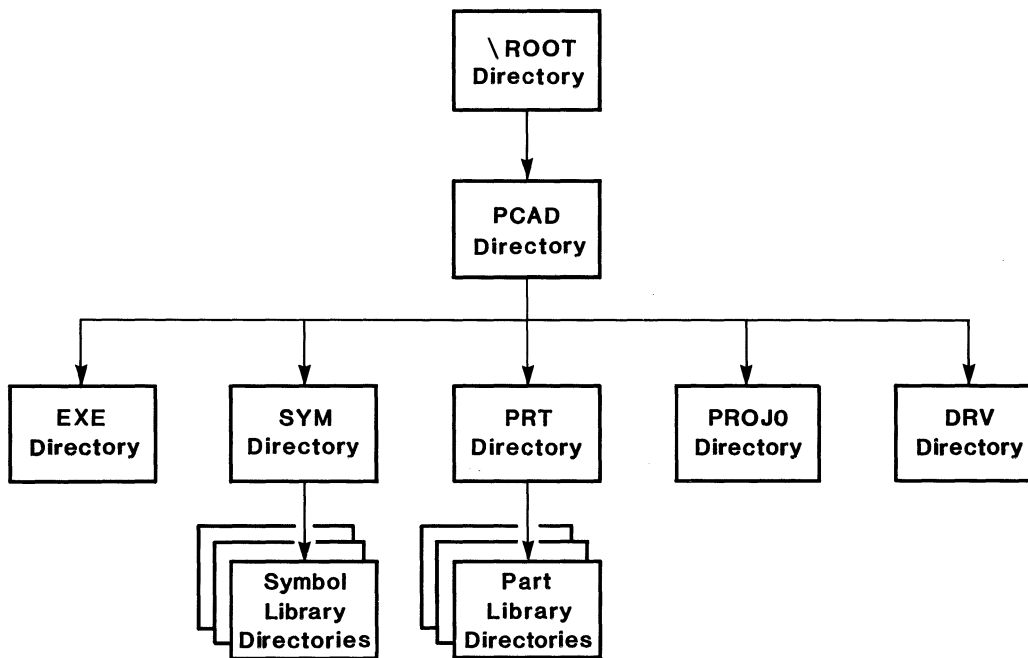
This chapter gives some basic operating instructions and provides an overview of the procedures to follow when you operate your system.

USING THE PROGRAMS

This section provides information on accessing your input and output files.

P-CAD Directory Structure

The P-CAD AUTOLOADER sets up the recommended directory structure and copies all the files into the appropriate directories as shown in Figure 3-1.



85029

Figure 3-1. P-CAD Recommended Directory Structure

For training examples, you should start from the \PCAD\PROJ0 directory. When using the system for your designs, you can create additional directories as necessary for groups of designs. P-CAD recommends that you start your programs from the current project directory so that the output of each program is in the appropriate directory to be the input to other programs.

Changing Directories

When you turn on your computer, the root directory is the current directory (unless you have set up your system to start from another directory). To change from the root directory to a subdirectory, or from a subdirectory to any other directory or subdirectory, type the DOS change directory command (CD) followed by a backslash and a complete path. For example, to change to the PCAD directory, at the C:\> prompt, type:

```
CD \PCAD [Return]
```

To change to the EXE subdirectory, type:

```
CD \PCAD\EXE [Return]
```

To change to the appropriate directory for performing the training examples, type:

```
CD \PCAD\PROJ0 [Return]
```

Refer to your DOS manual for additional information on changing directories.

Indicating Paths

P-CAD programs stored in one directory or drive may have to read from or write to files stored in another directory or drive.

If the program you are running displays a prompt for file input or output, and the necessary file is not in the current directory or drive, you must specify a path or drive name to the directory or drive where that file is located.

For instance, a P-CAD program might display the following prompt for a filename:

```
FILENAME : <filename>
```


You would respond in one of the following ways, depending on the location of the file.

- If the necessary file is in the current drive, simply type the filename and extension. For example:

FILENAME : TESTFILE.SCH

- If the file is in a subdirectory of the current directory, specify the subdirectory name. For example:

FILENAME : DATA\TESTFILE.SCH

- If the file is in another directory that is not a subdirectory of the current one, type in the full path name. For example:

FILENAME : \PCAD\DATA\TESTFILE.SCH

- If the necessary file is located on a disk in a different drive, type in the drive designator of the file. For example:

FILENAME : A:TESTFILE.SCH

Viewing and Printing ASCII Files

Several of the PCB programs produce ASCII files containing program operation logs, error messages, and design data for user reference.

The programs and their ASCII outputs are listed in Table 3-1. The error message files are produced only when the program encounters errors during operation.

Table 3-1. ASCII Output Files

Program	ASCII Output
PC-CAPS	Command log file (PCCAPS.CMD)
PC-CARDS	Command log file (PCCARDS.CMD) Swap report (<filename>.RPT)
PC-DRC/NLC	Error message file (<filename>.DRC)
PC-LINK	Message file (PCLINK.MSG)
PC-FORM	Nodes list (<filename>.NDE) Component list (<filename>.CMP) Packaging list (<filename>.PKL) Wiring list (<filename>.WRL) Bill of materials (<filename>.MAT)
PREPACK	Error message file (PREPACK.ERR)
PC-PACK	Packaging command file (<filename>.CMD) Error message file (PCPACK.ERR)
PC-BACK	Back-annotation file (<filename>.BKA) Swap report listing file (<filename>.LIS)
PC-PLACE	Command log file (PCPLACE.CMD) Swap report (<filename>.RPT) Histogram statistics report (<filename>.HIS) Automatic placement report (<filename>.PLR)
PC-ROUTE	Status report file (<filename>.REP)

You can view or print these files using a text editing program or using the DOS commands TYPE and PRINT.

To view or print your ASCII files using the DOS commands, first change to the directory containing the output file you want to view or print.

To view the file, use the DOS TYPE command:

```
TYPE <filename.ext> [Return]
```

The system scrolls the specified file on the screen. You can press [Ctrl]-[S] to start and stop automatic scrolling.

To print a hard copy of the file, use the DOS PRINT command:

```
PRINT <filename.ext> [Return]
```

The first time you use the PRINT command after each time you boot up your system, the system prompts:

```
Name of List Device? PRN
```

Accept the PRN default by pressing [Return], or enter LPT1, LPT2, or LPT3 to specify the port to which the printer is connected and press [Return]. The system prints the file.

SYSTEM CONCEPTS

This section introduces concepts that are used in computer aided design systems. For additional information on how to work with grid structure and layers, refer to the *PC-CAPS* and *PC-CARDS User's Manuals*.

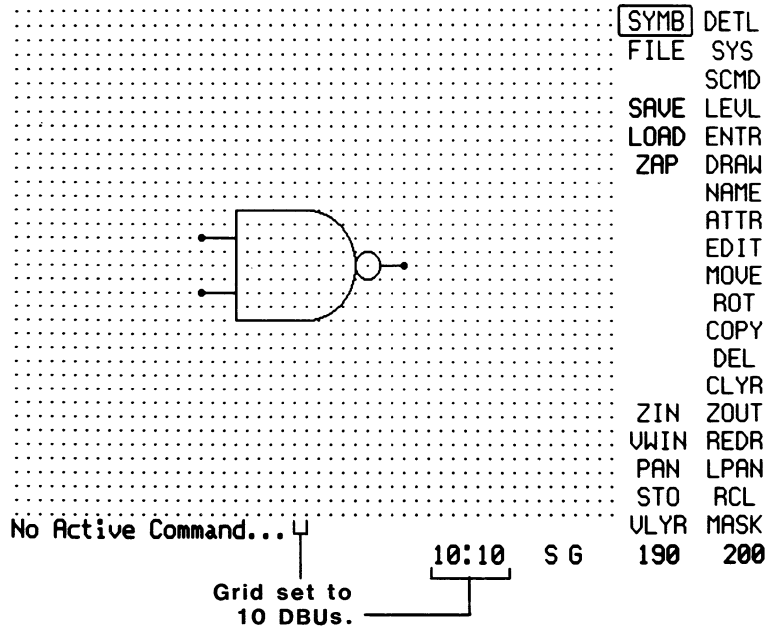
Grid Structure

In PC-CAPS, PC-CARDS, and PC-PLACE, the visible screen contains rows and columns of dots. These dots show the grid and they function like graph paper as a design aid for scale and sizing.

The scale of a design is based on a hypothetical database unit (DBU). You can define the grid size in database units using a status line command. The grid coordinates parameter is in the format X:Y, where X defines the distance in DBUs between two horizontal grid points and Y defines the distance in DBUs between two vertical grid points. The default grid coordinates are 10:10 in PC-CAPS and 50:50 in PC-CARDS, so the default distance between two grid points is 10 DBUs in PC-CAPS and 50 DBUs in PC-CARDS.

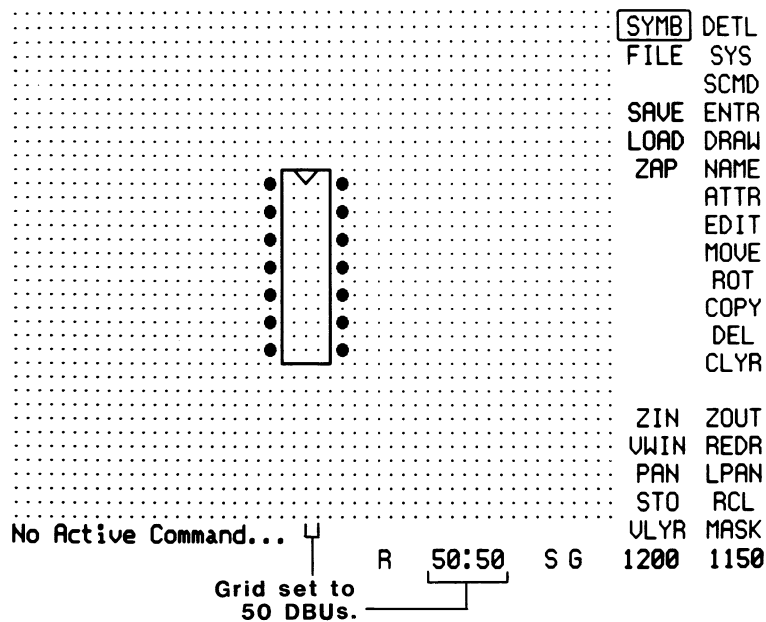
When you are working on a design, you can turn the visible grid display off and on at any time. You can also zoom in and zoom out to display a larger view of the design or a more detailed view of a particular area. As you zoom in, the grid points appear farther apart. As you zoom out, the points appear closer together. When you zoom out to a view in which the points would appear too close together, the grid points disappear.

You can change the grid coordinates parameter at any time; for example, you might change the grid coordinates from 50:50 to 100:100 because you do not need such a fine grid. However, you must be sure to use a standard scale for all symbols or parts you create, so that all the components in a design or PCB are drawn to the same scale. The P-CAD library symbols are drawn to the scale illustrated by the NAND symbol in Figure 3-2 and the 14-pin DIP symbol in Figure 3-3.



85030

Figure 3-2. Grid Scale of P-CAD Library Schematic Symbols



85036

Figure 3-3. Grid Scale of P-CAD Library Parts

The total area available for any design is 60,000 by 60,000 DBUs. The area visible on the screen is a portion of the total area. You can display a "long pan" view of a design to show the position of the screen you are working on in relationship to the area available for the design.

With PC-CAPS, in relation to a printer or plotter, when you use a scale of 1, one DBU equals 0.01 inch (10 mils). If you use the default PC-CAPS grid coordinate parameter of 10:10 and plot a schematic at a scale of 1, the distance between grid points is 0.1 inch.

With PC-CARDS, in relation to a printer or plotter, when you use a scale of 1, one DBU is equal to 0.001 inch (1 mil). If you use the default PC-CARDS grid coordinate parameter of 50:50 and plot a PCB design at a scale of 1, the distance between grid points is 0.05 inch.

The printing and plotting programs allow you to scale the schematic or PCB drawing to the size of your paper, so you can easily use a different scale on any plot.

Layer Structure

A schematic or PCB database is composed of several types of objects and information, such as component symbols, wires, pins, and descriptive text. The PCB systems allow you to assign each type of data to a separate graphic "layer". You can think of layers as clear plastic sheets placed together to create a drawing. P-CAD provides sets of predefined layers for schematic and PCB databases, which are recommended as a standard.

Each symbol, part, schematic, or PCB file has a set of layers; you can use the standard set or define your own. Symbols and parts in the P-CAD libraries are built using the standard layer structure, so if you use P-CAD library symbols and parts, you must use the standard layers. You can add new layers to a file at any time. Each layer has a name and a color, both of which you can change at any time. You can set any layer to be visible or not visible at any time.

The layer structure provides a way to include only certain types of schematic or PCB data in a printout or plot and to select the plot color for each type of data. You can turn layers on and off to determine what types of data are plotted. When you make a plot file, it includes only the data that is currently visible. For a pen plot, the number of the color you select for a layer determines which pen is used.

BACKING UP YOUR DATA FILES

You should back up your data files regularly by copying them onto floppy diskettes. If a data file is damaged and becomes unusable due to power failure or another problem, you can then use the most recent backup copy.

You should back up all your schematic and PCB data files and all the symbol and part files you have created. You may also want to back up plot files, netlists, and other data files. You may want to back up daily, weekly, or on another schedule.

When deciding which files to back up and how often to back up, keep in mind that if your working file is damaged, you will have to re-enter all the data that was entered since the last time you backed up that file. If a program output file such as a netlist or a packaged database is damaged and you have no backup, you will have to go through all the steps necessary to recreate the file.

When backing up, you need to back up only the files that have been altered since the last time you backed up.

To back up your files, first insert a blank formatted diskette into the floppy disk drive of your computer. For each file you want to back up, make sure you are in the appropriate directory, then type:

```
COPY <filename.ext> A: [Return]
```

Label each backup diskette clearly with the date and time and a name for the data.

It is a good idea to keep three "generations" of backup copies. That way, if you need to use a backup and the most recent one is damaged or lost, you have an earlier one. Use one diskette the first time you back up, a second diskette the second time, and a third diskette the third time. The fourth time you back up, reuse the first diskette (erasing the old first backup); the fifth time, reuse the second one; and so on. You can keep using the same three diskettes indefinitely in this manner.

You may also want to put a write protect tab on your backup disks right after you back up. Do not remove the tab until you are ready to re-use the disk.

You may want to keep two sets of backup copies, in two different locations, of some or all of your files.

If you need to use a backup copy of a file, first make sure you are in the appropriate directory. Insert the most recent backup diskette into the floppy disk drive, then copy the file by typing:

```
COPY A:<filename.ext>
```

Note that if you want to keep the damaged file, you should rename it; otherwise, it will be overwritten by the backup copy.

If backing up requires more than one diskette, you may want to use the DOS BACKUP and RESTORE commands instead of the COPY command. Refer to your DOS manual for more information.

PROCEDURES FOR END-TO-END SCHEMATIC TO PCB DESIGN

When you are ready to build your own design, use the following steps as guidelines. Some of the steps are optional, and you may want to use some of the programs in a different order.

1. If you plan to use an interface to another system, read the user's manual for the interface program, paying special attention to the "Preparing Your Schematic Database" section.
2. Use PC-CAPS symbol mode to build any symbols that are not included in your symbol library and that you need for the schematic.
3. Use PC-CARDS symbol mode to construct any part files that are not included in your parts libraries and that you need for the PCB. If you plan to photoplot your PCB, use PC-CARDS symbol mode to construct any padstack symbols you need for photoplotting, then use a text editor to create the special symbol file.

4. Use a text editor to create or edit the ASCII component library cross-reference file, then use PREPACK to convert it to binary format.
5. Use PC-CAPS detail mode to construct the schematic. If necessary, use PC-CAPS symbol mode to draw a symbol for the schematic and/or to assign attributes.
6. Use PC-PRINT or PC-PLOTS to produce a printout or plot of the schematic.
7. Use PC-NODES to extract the schematic netlist. If the schematic is multisheet or hierarchical, extract a netlist for each schematic database file in the design.
8. If the schematic design is multisheet or hierarchical, use PC-LINK to link the netlists.
9. Use PC-FORM to produce ASCII-format listings for the schematic.
10. Use PC-PACK to package the schematic into a PCB database. If you want to see netlists for the PCB database, input the packaging netlist created by PC-PACK into PC-FORM. If you want to update the schematic before making further changes to the PCB design, use PC-CAPS to input the packaging command file created by PC-PACK into the schematic database.
11. Use PC-CARDS or PC-PLACE to determine final placement for the components on the PCB design. With PCB-3 only, use PC-CARDS or PC-PLACE to add routing barriers for use with PC-ROUTE. To pre-wire traces for PC-ROUTE, you must use PC-CARDS.
12. (PCB-3 only) Use PC-ROUTE to automatically route the PCB.
13. Use PC-CARDS to enter the traces on the PCB design. (With PCB-3, enter only the traces that PC-ROUTE was unable to complete.)
14. Use PC-DRC/NLC to check the PCB database for violations of design rules. If necessary, use PC-CARDS to correct design errors.

15. Input the packaging command file produced by PC-PACK and the swap report file produced by PC-CARDS or PC-PLACE into PC-BACK to produce the back-annotation file. Input the back-annotation file into PC-CAPS to update the schematic.
16. If you intend to photoplot the PCB design, use PC-CARDS to enter padstacks by calling up the special symbol file.
17. Use PC-PRINT, PC-PLOTS, or PC-PHOTO to produce the final plot of the PCB.

APPENDIX A. PCB SYSTEM MATERIALS

The package for each PCB system includes several diskettes and corresponding user's manuals.

The diskettes contain the following:

Programs:

- PC-CAPS (PCB-2 and PCB-3 only)
- PC-CARDS
- PC-LINK
- PC-NODES
- PC-FORM
- PREPACK
- PC-PACK
- PC-BACK-
- PC-PRINT
- PC-PLOTS
- PC-PHOTO
- PC-DRC/NLC
- PC-PLACE (PCB-3 only)
- PC-ROUTE (PCB-3 only)

Sample Data:

- PC-CAPS Examples (PCB-2 and PCB-3 only)
- PC-CARDS Examples

Libraries (one or more if ordered):

TTL Schematic Symbols Library
TTL PCB Parts Library

CMOS Schematic Symbols Library
CMOS PCB Parts Library

Microprocessor Schematic Symbols Library
Microprocessor PCB Parts Library

Discrete Schematic Symbols Library
Discrete PCB Parts Library

Linear Standard Schematic Symbols Library
Linear Standard PCB Parts Library

The package also includes the following manuals:

P-CAD Installation Guide
PCB System Overview (this manual)
PC-CAPS User's Manual (PCB-2 and PCB-3 only)
PC-CARDS User's Manual
PC-PLACE and PC-ROUTE User's Manuals (PCB-3 only)
Utilities User's Manuals
Library Manual(s) (if libraries ordered)

APPENDIX B. P-CAD FILENAME EXTENSION CONVENTIONS

P-CAD programs create several types of files. For some files, the program allows you to specify the filename; for others, the program assigns the name, using the database filename and assigning a new extension. When you specify the filename, P-CAD recommends that you use the P-CAD conventions for filename extensions. It is also important to use letters and numbers only in filename extensions; some programs in the system do not recognize certain special characters such as percent (%) and underscore(_).

Table B-1 gives the filename extensions used in the PCB systems. P-CAD's interface programs and the programs in P-CAD's CAE systems use additional extensions not listed here. PC-PRINT, PC-PLOTS, and PC-PHOTO can produce files for specific devices; the filename extensions used with these files are listed in Table B-2.

Table B-1. PCB Filename Extension Conventions

Extension	Description	Produced By
.BKA	Back-annotation command file	PC-BACK
.BNL	Back-annotated netlist file	PC-PACK
.CFG	Configuration file	PC-CAPS PC-CARDS PC-PRINT PC-PLOTS
.CMD	Packaging command file	PC-PACK
.CMD	Command log file	PC-CAPS PC-CARDS PC-PLACE PC-FORM
.CMP	Component list	PC-FORM

Table B-1 Continued

Extension	Description	Produced By
.CTL	Routing strategy file	PC-ROUTE
.DBG	Memory allocation file	PC-CAPS PC-CARDS
.DRC	Error message file	PC-DRC/NLC
.ERR	Error message file	PREPACK PC-PACK
.FIL	ASCII cross-reference file	User
.HIS	Histogram statistics report	PC-PLACE
.LIB	Binary cross-reference file	PREPACK
.LIS	Swap report listing file	PC-BACK
.MAT	Materials list	PC-FORM
.MSG	Message file	PC-LINK
.NDE	Node list	PC-FORM
.NET	Verified netlist file	PRESIM
.NLT	Schematic netlist file	PC-NODES
.OUT	Command log file	PC-PLACE
.PCB	PCB database file	PC-PACK PC-CARDS PC-DRC/NLC PC-ROUTE
.PKL	Packaging list	PC-FORM
.PLC	Placed PCB database	PC-PLACE
.PLR	Automatic placement report file	PC-PLACE
.PLT	Plot file	PC-CAPS PC-CARDS PC-PLACE
.PNL	Physical netlist file	PC-PACK PC-NODES
.PRT	PCB part file	PC-CARDS
.PS	Padstack file	PC-CARDS
.RPT	Swap report file	PC-CARDS PC-PLACE
.REP	Status report file	PC-ROUTE
.SCH	Schematic database file	PC-CAPS

Table B-1 Continued

Extension	Description	Produced By
.SDT	Solder dot file	PC-CAPS
.SSF	Special symbol file	User
.SYM	Schematic component symbol	PC-CAPS
.WRL	Wire list	PC-FORM
.XNL	Expanded schematic netlist file	PC-LINK

Table B-2. PC-PRINT, PC-PLOTS, and PC-PHOTO Filename Extension Conventions

Extension	Description	Produced By
.APR	Aperture listing	PC-PHOTO
.CC	Calcomp penplotter format plot file	PC-PLOTS
.CTH	C.Itoh printer format plot file	PC-PRINT
.DMP	Houston Instruments (Bausch & Laumb) format plot file	PC-PLOT
.DSN	Seiko D-SCAN format plot file	PC-PLOT
.EPS	Epson format plot file	PC-PRINT
.FLS	Flashscan format photoplot file	PC-PHOTO
.GBR	Gerber format photoplot file	PC-PHOTO
.GTC	GTCO format photoplot file	PC-PHOTO
.GTK	Graphtec format plot file	PC-PLOTS
.HP	Hewlett-Packard format plot file	PC-PLOTS
.IBM	IBM format plot file	PC-PLOTS
.MTH	Mutoh format plot file	PC-PLOTS
.NZ	Nicolet Zeta format plot file	PC-PLOTS
.OKI	Okidata format plot file	PC-PRINT

APPENDIX C. LOADABLE DEVICE DRIVER FILES

This appendix describes how to create the P-CAD loadable driver reference file and lists the device driver files provided by P-CAD.

THE LOADABLE DRIVER REFERENCE FILE

The P-CAD loadable driver reference file (PCADDRV.SYS) configures the P-CAD software to your computer hardware.

If your system does not already have a loadable driver reference file containing the appropriate driver files, you must create one. You can create this file using a text editor or the COPY command.

NOTE: If you have installed your software using the P-CAD AUTOLOADER, you do not need to create the system driver file. The INSTALL program has already created it with the appropriate driver files. However, if desired, you can edit this file.

To create the loadable driver reference file with a text editor, follow the steps below.

1. Create a file named PCADDRV.SYS and type:

```
SYSTEM \PCAD\DRV\DISPLAY \PCAD\DRV\INPUT \PCAD\DRV\
```

The first line specifies the path and name of the system driver file for your computer system.

The second line specifies the path and name of the display driver file for your graphics adapter and monitor.

The third line specifies the path and name of the input device driver file for your input device type.

Filenames with a .DRV extension are device driver files, which the AUTOLOADER program copies into the \PCAD\DRV directory. The following sections provide a list of the driver files provided by P-CAD.

NOTE: You can also add a line to the loadable driver reference file to change the default colors. Refer to Appendix E, "Color Selection," in the *PC-CAPS User's Manual* or Appendix D, "Color Selection," in the *PC-CARDS User's Manual*.

2. Copy the file you created to the root directory by typing:

```
COPY PCADDRV.SYS C:\ [Return]
```

To create the system driver file with the COPY command:

1. At the root directory, type:

```
COPY CON: PCADDRV.SYS [Return]
SYSTEM \PCAD\DRV\.DRV [Return]
DISPLAY \PCAD\DRV\.DRV [Return]
INPUT \PCAD\DRV\.DRV [Return]
```

2. Press [Ctrl]-[Z] or [F6], and then press [Return] to save the information that you entered. The system responds with the message:

```
1 file(s) copied
```

SYSTEM DRIVER FILES

Following is a list of the system driver files provided by P-CAD, their filenames, and any special hardware or software requirements needed to use them.

IBM PC, PC/XT, PC/AT

Name: **SIBMPC.DRV**

Requirements: IBM PC Async card.

Notes: The driver interfaces directly to the hardware and does not use any BIOS calls for the serial ports. This means that the driver will only work for systems which have the 8250 chip with the IBM port addresses. The serial port may have 9-pin or 25-pin male connectors, depending on which vendor's Async card is installed.

Tandy TRS-80 Model 2000 Personal Computer

Name: **STRSPC.DRV**

Requirements: Tandy TRS-80 Model 2000 PC.

Notes: The driver uses the Tandy BIOS routines for serial communication. Because of the 8251 chip, the Tandy driver will not transmit characters unless the hardware is handshaking with the clear to send (CTS) line. This means that when the hardware does not provide this type of handshaking, a special cable is required. The P-CAD security device needs this special cable for the Tandy machines. The Tandy 2000 has a female 25-pin connector. The special cable should tie together pins 4 and 5 (RTS and CTS) on the computer side. When operating a device that needs handshaking (for example, a plotter) you must remove the special cable.

Texas Instruments Professional Computer

Name: STIPC.DRV

Requirements: Texas Instruments Professional Computer.

Notes: The driver allows up to four serial ports. These ports are the standard TI ports with the Z8530 chip. The TI machine has female 25-pin connectors.

DISPLAY DEVICE DRIVER FILES

Following is a list of the display device driver files provided by P-CAD, their filenames, and any special hardware or software requirements needed to use them.

AT&T 6300 (see Olivetti)

BNW Precision driver - 1024 x 768

Name: DBNW.DRV

Requirements: BNW Precision Graphics Adapter

Notes: A line <path> GRAPH should be added to the AUTOEXEC.BAT file to execute the program GRAPH.EXE (supplied by BNW) to initialize the adapter for the monitor.

Conographics driver - 640 x 400, 16 colors

Name: DCONO.DRV

Requirements: Conographic Corp. Model 40 graphics adapter with an analog monitor.

Notes: The driver requires the MODEL40.SYS driver (supplied by Conographics) to be resident in memory. This is accomplished by adding the following line to your CONFIG.SYS file:

DEVICE = MODEL40.SYS <parameter list>

The <parameter list> for the Electrohome is:

/FAST/ANALOG/EXTENDED/CGSET/260/200

Galaxy 819 driver - 800 X 600

Name: DGAL819.DRV

Requirements: Galaxy sextant-819 adapter and monitor.

Notes: A line <path> GALSTART should be added to the AUTOEXEC.BAT file to execute the program GALSTART.EXE (supplied by TAT Graphics) to initialize the adapter for the monitor.

Galaxy 1019 driver - 1024 x 768

Name: DGAL1019.DRV

Requirements: Galaxy sextant-1019 adapter and monitor.

Notes: A line <path> GALSTART should be added to the AUTOEXEC.BAT file to execute the program GALSTART.EXE (supplied by TAT Graphics) to initialize the adapter for the monitor.

Hercules driver - 720 x 348, 2 colors (black & white)

Name: **DHERCUL.DRV**

Requirements: Hercules monochrome graphics board and IBM (or compatible) monochrome monitor.

IBM CGA driver - 320 x 200, 4 colors

Name: **DIBMCGA.DRV**

Requirements: IBM (or compatible) Color Graphics Adapter and IBM PC Color Display.

IBM EGA driver - 640 x 350, 16 colors

Name: **DIBMEGA.DRV**

Requirements: IBM Enhanced Graphics Adapter with the Graphics Memory Expansion Board containing 64K, for a total of 128K of memory, and an IBM Enhanced Color Display.

Notes: When using the EGA on a PC or PC/XT, switches 5 and 6 of switch block 1 on the motherboard must be in the ON position.

Jumper P1 on the EGA board must be placed on pins 1 and 2.

Jumper P3 on the EGA board must be placed on pins 1 and 2.

IBM PGC driver - 640 x 480, 16 colors

Name: **DIBMPGC.DRV**

Requirements: IBM Professional Graphics Controller and Professional Graphics Display.

Metheus Omega PC driver - 1024 x 768

Name: DMETHEUS.DRV

Requirements: Metheus Omega-PC adapter and monitor.

Notes: The line OMEGAUEX OMEGA_PC.HEX
OMEGA_PC.FNT should be added to the
AUTOEXEC.BAT file to execute the program
OMEGAUEX.EXE (supplied by Metheus) to download
the microcode (OMEGA_PC.HEX) and default font
(OMEGA_PC.FNT) to the adapter.

Olivetti (AT&T) driver - 640 x 400, 16 colors

Name: DOLIVET.DRV

Requirements: Olivetti color graphics board with Enhanced Graphics
Color Board (EGC) attached and AT&T Color Monitor.

P-CAD HR-25 driver - 1024 x 800, 16 colors

Name: DHR25.DRV

Requirements: P-CAD HR-25 graphics adapter and high resolution
monitor running on an IBM PC/AT.

Notes: The driver program assumes the graphics microcode
file, MVDI.LOD, is located in the \PCAD\DRV
subdirectory.

Tandy TRS-80, Graphics Monitor driver - 640 x 400, 8 colors

Name: DTRS.DRV

Requirements: Tandy color graphics adapter and High Resolution
Color Monitor CMI.

Texas Instruments Professional Color Monitor driver - 720 x 300, 8 colors

Name: DTIPC.DRV

Requirements: Texas Instruments Color Graphics Controller and Color Monitor.

Vectrix Graphics driver - 1024 x 1024, 16 colors

Name: DVECTRIX.DRV

Requirements: Vectrix PEPE graphics controller and a monitor with the following characteristics:

1024 x 1024 resolution
60 Hz non-interlaced
100+ MHz video rate
64 KHz horizontal scan rate
Separate horizontal & vertical sync

IBM CGA and color display monitor are also required because this is a two-monitor system.

Notes: The following lines must be added to the AUTOEXEC.BAT file:

PEPINT <monitor parameter>
RESTART

PEPINT.EXE and RESTART.EXE are initialization programs to initialize the PEPE board.

The PEPINT.EXE requires a monitor parameter file. When using the MONITRONIX monitor, the parameters are in the file MONITRON. The line would be:

PEPINT MONITRON

These files must be in the root directory or a path must be included to where they are located.

INPUT DEVICE DRIVER FILES

Following is a list of the input driver files provided by P-CAD, their filenames, and any special hardware or software requirements needed to use them.

GTCO Pad Digitizer

Name: IGTCO.DRV

Requirements: GTCO digitizer.

Notes: The driver supports either the five-button or the one-button version. The driver will not work with the 16 button version. All sizes are supported. The driver uses 6 byte packed binary format at 1200 baud, 8 data bits, no parity, and 1 stop bit. The digitizer requires a cable which allows handshaking. A NULL-MODEM cable will suffice.

Houston Instruments True Grid 1011 and HIPAD Digitizer

Name: 1HIPAD.DRV

Requirements: Houston Instruments True Grid 1011 or HIPAD digitizer.

Notes: The driver supports the True Grid 1011 and HIPAD four-button puck at 4800 baud, 8 data bits, no parity, and one stop bit.

Logitech Mouse (see Mouse Systems Mouse)

Mouse Systems Mouse

Name: **IMOUSYS.DRV**

Requirements: **Mouse Systems Mouse or compatible. Serial Port.**

Texas Instruments Mouse

Name: **ITI.DRV**

Requirements: **TI Mouse and TI Mouse interface board.**

Notes: **This mouse is a special Mouse Systems Mouse with a 9-pin connector plugged into a special board on the TI which has an 8251 USART.**

INDEX

A

AUTOLOADER, 1-3, 3-1, C-1

B

Back-annotation, 2-2, 2-9, 2-11, 2-14

Back-annotation file, 2-14, 3-13

Backing up, 3-11, 3-12

C

Component library cross-reference file, 2-8, 3-13

D

DBU (database unit), 3-7, 3-9

Directory

 changing, 3-3

 indicating path, 3-3

 P-CAD structure, 3-1

F

Filenames, 2-1

 extensions, 2-1, B-1

Files, 2-1, B-1

 ASCII, 3-4

 backing up, 3-11, 3-12

 input and output for programs, 2-18

 printing, 3-4, 3-6

 viewing, 3-4, 3-6

Force vectors, 2-12

G

Grid structure, 3-6, 3-7

H

Histogram, 2-12

I

INSTALL program, 1-3, C-1

Installing the system, 1-1

Interfaces to other CAD systems, 1-1, 2-7, 3-12

L

Layer structure, 3-10

Library, 2-9, A-1

part, 2-11

symbol, 2-2

Loadable device driver files, 1-3, C-1

N

Netlists, 2-2, 2-7, 2-9, 2-11, 2-14

ASCII, 2-7, 2-14

backing up, 3-11

binary, 2-7, 2-8, 2-14, 3-13

expanded, 2-7

P

- Packaged netlist file, 2-9
- Packaging command file, 2-9, 2-14, 3-13
- Packaging schematics, 2-2, 2-8, 2-9
- Padstacks, 3-12, 3-14
- Parts, 2-9, 2-12
 - creating, 2-11
 - layers of, 3-10
 - scale of, 3-7
- PC-BACK, 2-9, 2-14
- PC-CAPS, 2-1, 2-2, 2-4, 2-5, 2-8, 2-9, 3-12, 3-13
 - and printing schematics, 3-9
 - detail mode, 2-3, 2-4
 - grid display, 3-7
 - symbol mode, 2-2, 2-4
- PC-CARDS, 2-1, 2-8, 2-10, 2-11, 3-12, 3-13
 - and printing PCB designs, 3-10
 - detail mode, 2-11, 2-12
 - grid display, 3-7
 - symbol mode, 2-11
- PC-DRC/NLC, 2-13, 3-13
- PC-FORM, 2-9, 2-14, 3-13
- PC-LINK, 2-7, 2-8, 3-13
- PC-NODES, 2-7, 2-8, 3-13
- PC-PACK, 2-8, 2-9, 2-14, 3-13
- PC-PHOTO, 3-14
- PC-PLACE, 2-1, 2-12, 2-14, 3-13
- PC-PLOTS, 3-13, 3-14
- PC-PRINT, 3-13, 3-14
- PC-ROUTE, 2-1, 2-12, 3-13
- PCB database file, 2-9, 2-10, 2-11, 2-12, 3-10, 3-13
 - back-annotating changes to schematic, 2-14
 - backing up, 3-11
 - checking for errors, 2-13
 - creating from parts, 2-12
 - packaging a schematic into, 2-11, 2-12
 - part placement analysis, 2-12
 - photoplotting, 2-15
 - routed, 2-12

Plot files

- backing up, 3-11
- Plots, 2-2, 2-8, 2-11, 3-9, 3-10, 3-13
- pen, 2-8, 2-15
- photo, 2-8, 2-15, 3-14
- PREPACK, 2-8, 3-13
- PRINT command, 3-6
- Printouts, 2-2, 2-8, 2-11, 3-9, 3-10, 3-14
- dot-matrix, 2-8, 2-15

R

Routing

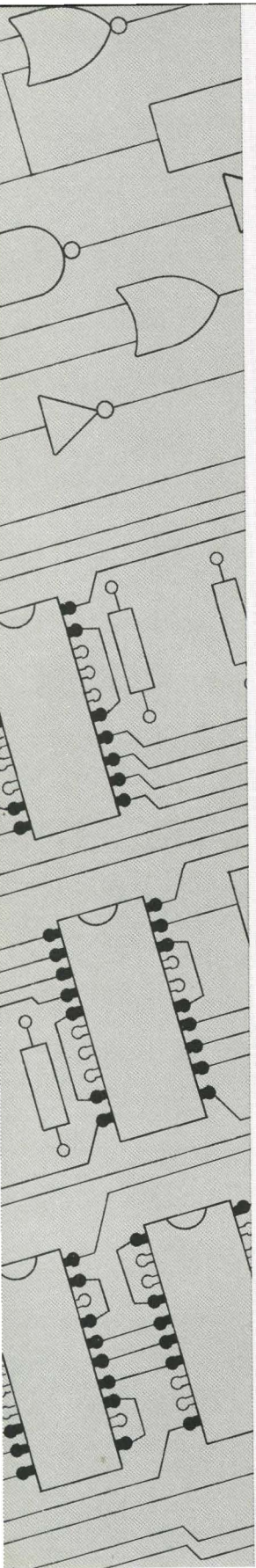
- automatic, 2-12, 3-13

S

- Schematic database file, 2-3, 2-4, 3-10, 3-12, 3-13
 - back annotating, 2-14
 - backing up, 3-11
- Schematics, 3-13
 - back annotating, 2-2
 - creating, 2-1, 2-2, 2-3
 - hierarchical, 2-1, 2-3, 2-4, 2-7, 3-13
 - multisheet, 2-1, 2-3, 2-5, 2-7, 3-13
 - packaging, 2-8
- Special symbol file, 3-12, 3-14
- Swap report file, 2-14, 3-14
- Symbols, 2-1, 2-9, 3-12
 - creating, 2-2
 - layers of, 3-10
 - representing schematics, 2-4
 - scale of, 3-7

T

- TYPE command, 3-6



000-0097-02

p-cad[®]
PERSONAL CAD SYSTEMS INC.